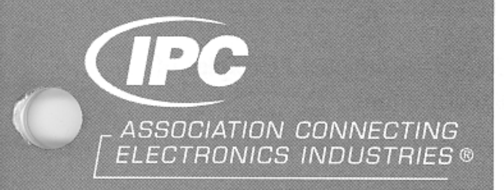


IPC-2223

Sectional Design Standard for Flexible Printed Boards

IPC-2223
November 1998

Supersedes IPC-D-249

A standard developed by IPC

2215 Sanders Road, Northbrook, IL 60062-6135
Tel. 847.509.9700 Fax 847.509.9798
www.ipc.org

The Principles of Standardization

In May 1995 the IPC's Technical Activities Executive Committee adopted Principles of Standardization as a guiding principle of IPC's standardization efforts.

Standards Should:

- Show relationship to Design for Manufacturability (DFM) and Design for the Environment (DFE)
- Minimize time to market
- Contain simple (simplified) language
- Just include spec information
- Focus on end product performance
- Include a feedback system on use and problems for future improvement

Standards Should Not:

- Inhibit innovation
- Increase time-to-market
- Keep people out
- Increase cycle time
- Tell you how to make something
- Contain anything that cannot be defended with data

Notice

IPC Standards and Publications are designed to serve the public interest through eliminating misunderstandings between manufacturers and purchasers, facilitating interchangeability and improvement of products, and assisting the purchaser in selecting and obtaining with minimum delay the proper product for his particular need. Existence of such Standards and Publications shall not in any respect preclude any member or nonmember of IPC from manufacturing or selling products not conforming to such Standards and Publication, nor shall the existence of such Standards and Publications preclude their voluntary use by those other than IPC members, whether the standard is to be used either domestically or internationally.

Recommended Standards and Publications are adopted by IPC without regard to whether their adoption may involve patents on articles, materials, or processes. By such action, IPC does not assume any liability to any patent owner, nor do they assume any obligation whatever to parties adopting the Recommended Standard or Publication. Users are also wholly responsible for protecting themselves against all claims of liabilities for patent infringement.

IPC Position Statement on Specification Revision Change

It is the position of IPC's Technical Activities Executive Committee (TAEC) that the use and implementation of IPC publications is voluntary and is part of a relationship entered into by customer and supplier. When an IPC standard/guideline is updated and a new revision is published, it is the opinion of the TAEC that the use of the new revision as part of an existing relationship is not automatic unless required by the contract. The TAEC recommends the use of the lastest revision. Adopted October 6. 1998

Why is there a charge for this standard?

Your purchase of this document contributes to the ongoing development of new and updated industry standards. Standards allow manufacturers, customers, and suppliers to understand one another better. Standards allow manufacturers greater efficiencies when they can set up their processes to meet industry standards, allowing them to offer their customers lower costs.

IPC spends hundreds of thousands of dollars annually to support IPC's volunteers in the standards development process. There are many rounds of drafts sent out for review and the committees spend hundreds of hours in review and development. IPC's staff attends and participates in committee activities, typesets and circulates document drafts, and follows all necessary procedures to qualify for ANSI approval.

IPC's membership dues have been kept low in order to allow as many companies as possible to participate. Therefore, the standards revenue is necessary to complement dues revenue. The price schedule offers a 50% discount to IPC members. If your company buys IPC standards, why not take advantage of this and the many other benefits of IPC membership as well? For more information on membership in IPC, please visit www.ipc.org or call 847/790-5372.

Thank you for your continued support.



IPC-2223

Sectional Design Standard for Flexible Printed Boards

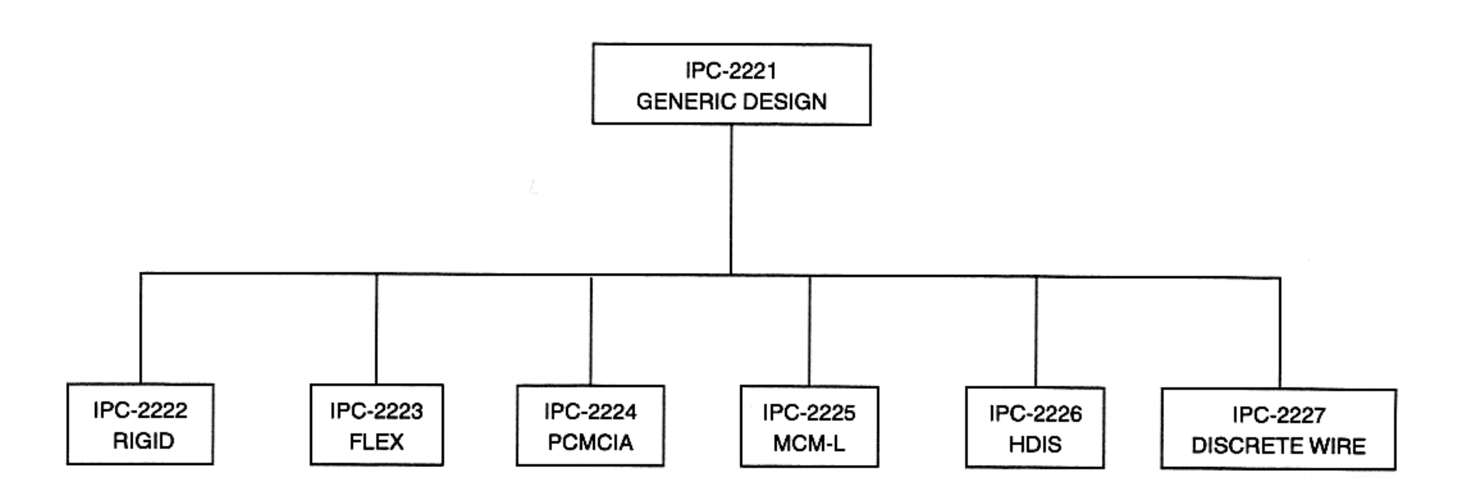
Developed by the Flexible Circuits Design Subcommittee (D-11) of the Flexible Circuits Committee (D-10) of IPC



Users of this standard are encouraged to participate in the development of future revisions.

Contact:

IPC
2215 Sanders Road
Northbrook, Illinois
60062-6135
Tel 847 509.9700
Fax 847 509.9798



Foreword

This standard is intended to provide information on the detailed requirements for flexible printed wiring design. All aspects and details of the design requirements are addressed in this sectional.

The information contained herein is intended to supplement generic design requirements identified in IPC-2221. This sectional standard, used in conjunction with IPC-2221, supersedes IPC-D-249.

Benefits of Flex

Due to the thin films used in flexible circuitry, flex can save weight and space and conform to three-dimensional configurations.

Printed circuits in flex can be fanned out to allow the use of different connectors and folded to change orientation.

A number of flexible dielectrics are available (e.g., polyimide, polyester, polyetherimide), some of which can offer thermal stability, while others can be used in low cost applications.

The thermal stability of polyimide flexible circuits allows the use of through hole and or surface mount components. Polyester circuits can offer lower cost circuits with limited component usage. Polyester is typically used for low soldering temperature applications.

The flexibility of these circuits can allow movement for ease in installation and/or maintenance during use.

A combination of flexible circuitry and rigid PWB technology can be combined for improved packaging in confined spaces through interconnect elimination/simplification.

Acknowledgment

Any Standard involving a complex technology draws material from a vast number of sources. While the principal members of the Flexible Circuits Design Subcommittee (D-11) of the Flexible Circuits Committee (D-10) are shown below, it is not possible to include all of those who assisted in the evolution of this standard. To each of them, the members of the IPC extend their gratitude.

Flexible Circuits Committee

Chairman
William Jacobi
Sheldahl, Inc.

Flexible Circuits Design Subcommittee

Chairman
Russell Griffith
Tyco PCG/Engineered Systems

Technical Liaison of the IPC Board of Directors

Peter J. Murphy
Parlex Corp.

Flexible Circuits Design Subcommittee

Gary Bachler, ADFlex Solutions, Inc.

Nanci Baggett, Dell, Inc.

Steve Bakke, Alliant Techsystems, Inc.

Rich Barnett, Compaq Computer Corp.

Frank Belisle, Sundstrand Aerospace

Robert Black, Northrop Grumman Electronic Sensors & Systems Div

Andrew Brannen, Litronics Corp.

Dennis Cantwell, Printed Circuits, Inc.

David Corbett, Defense Supply Center Columbus DSCC

Paul Deschamps, ADFlex Solutions, Inc.

Lawrence Dexter, Advanced Circuit Technology

Joe DiPalermo, Parlex Corp.

C. Don Dupriest, Lockheed Martin Vought Systems

David Fellows, Medtronic Inc/ Micro-Rel Division

Thomas Fisher, E. I. du Pont de Nemours and Co.

Thomas Gardeski, DuPont High Performance Films

Lyle Harford, Raytheon Systems Co.

Gerald Heppell, Hewlett Packard Co.

Phillip Hinton, Hinton 'PWB' Engineering

Robert Hodges, Rogers Corp.

M. Frederick Hoover, Polyonics Corp.

Allen Hullenaar, Motorola GSTG

Dan Jacobus, Sheldahl, Inc.

Dennis Johnson, Packard Hughes Interconnect

Roy Keen, Rockwell International

Richard King, Honeywell, Inc./Air Transport Systems

Clarence Knapp, Litton Guidance & Control Systems

Nick Koop, Minco Products, Inc.

George Kotecki, Northrop Grumman Corp.

Thomas Kurtz, Raytheon Systems Co.

Shawn Lien, Ciba Speciality Chemicals Corp.

Doug Miller, Rockwell International

Dominique Numakura, Litchfield Precision Components

William Ortloff, Raytheon Systems Co.

Richard Peyton, Lockheed Martin Astronautics

Bruce Rietdorf, Raytheon Systems Co.

Joseph Schmidt, Raytheon Systems Co.

Kelly Schriver, SCI Systems, Inc.

Robert Sheldon, Packard Hughes Interconnect

Grant (Rick) Smedley, Raytheon Systems Co.

Clark Webster, General Dynamics Information Systems, Inc.

Robert Wieder, DuPont High Performance Films

John Williams, Raytheon Co.

Chester Wright, Teledyne Electronic Technologies

Fonda Wu, Raytheon Systems Co.

Richard Zwierlein, E. I. du Pont de Nemours and Co.

Table of Contents

Figures

Tables

This Page Intentionally Left Blank

Sectional Design Standard for Flexible Printed Boards

1 SCOPE

This standard establishes the specific requirements for the design of flexible printed wiring applications and its forms of component mounting and interconnecting structures. The flexible materials used in the structures are comprised of insulating films, reinforced and/or non-reinforced, dielectric in combination with metallic materials. These interconnecting boards may contain single, double, multi-layer, or multiple conductive layers and can be comprised wholly of flex or a combination of both flex and rigid.

1.1 Purpose The requirements contained herein are intended to establish specific design details that **shall** be used in conjunction with IPC-2221.

1.2 Classification of Products Classification of products **shall** be in accordance with IPC-2221 and as stated in 1.2.1 and 1.2.2.

1.2.1 Board Type

Type 1 Single-sided flexible printed wiring containing one conductive layer, with or without stiffener (see Figure 1-1)

Type 2 Double-sided flexible printed wiring containing two conductive layers with plated-through holes, with or without stiffeners (see Figure 1-2)

Type 3 Multilayer flexible printed wiring containing three or more conductive layers with plated-through holes, with or without stiffeners (see Figure 1-3)

Type 4 Multilayer rigid and flexible material combinations containing three or more conductive layers with plated-through holes (see Figure 1-4)

Type 5 Flexible or rigid-flex printed wiring containing two or more conductive layers without plated-through holes (see Figure 1-5)

1.2.2 Installation Uses Flexible circuit designs are unique in each application; however, the following are some typical classes of use. It is recommended that the intended use be specified on the fabrication drawing. It may be necessary to define specific tests for design verification on the master drawing. These categories can be used individually or in a combination.

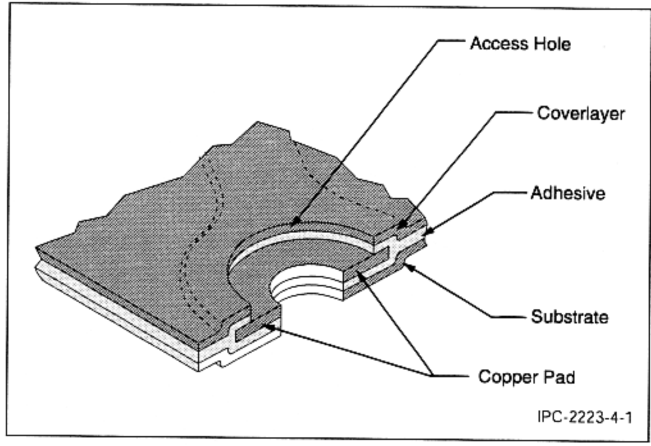


Figure 1-1 Board Type 1

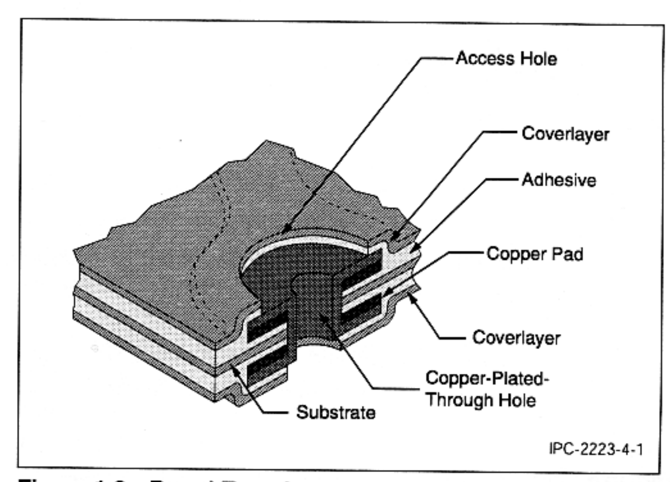


Figure 1-2 Board Type 2

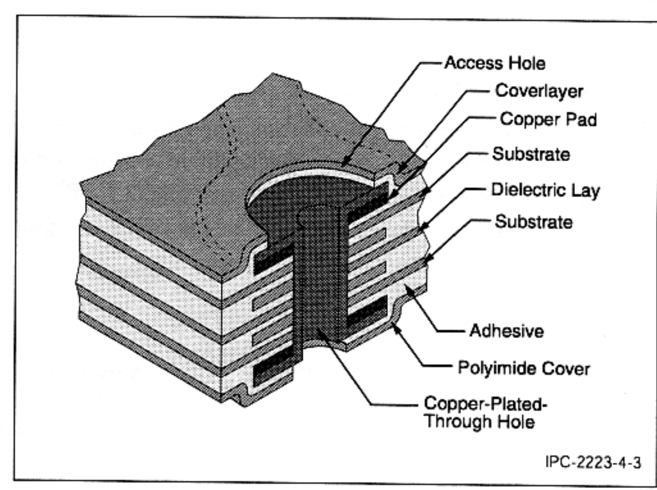


Figure 1-3 Board Type 3

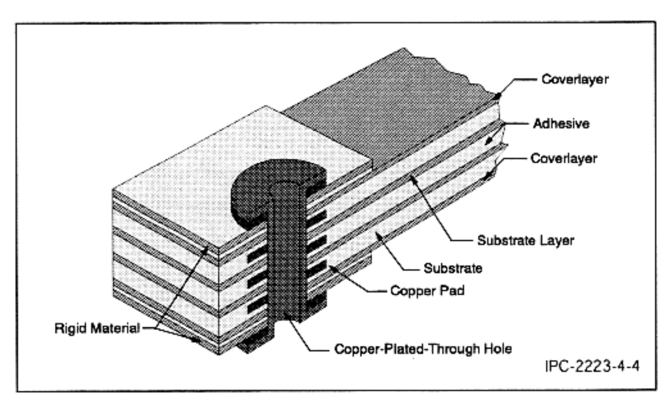


Figure 1-4 Board Type 4

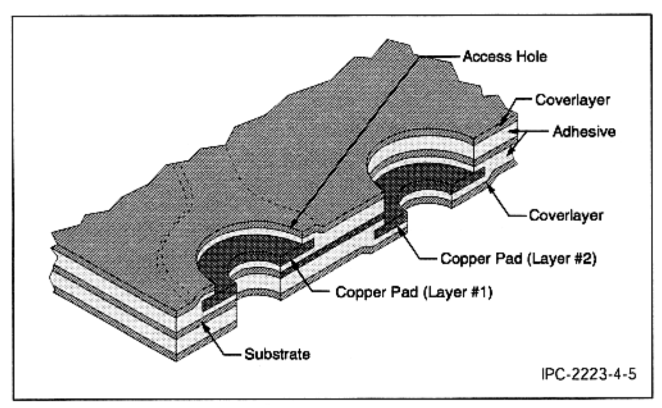


Figure 1-5 Board Type 5

Use A Capable of withstanding flex during installation (flex-to-install)

Use B Capable of withstanding continuous flexing for the number of cycles as specified on the master drawing (dynamic flex)

Use C High temperature environment (over 105°C)

Use D UL recognition

2 APPLICABLE DOCUMENTS

The following documents form a part of this standard to the extent specified herein. If a conflict of requirements exist between IPC-2223 and those listed in 2.1, IPC-2223 takes precedence. The revision of the document in effect at the time of solicitation **shall** take precedence.

2.1 IPC[1]

IPC-MF-150 Metal Foil for Printed Wiring Applications

IPC-FC-231 Flexible Bare Dielectrics for use in Flexible Printed Wiring

IPC-FC-232 Adhesive Coated Dielectric Films for Use as Cover Sheets for Flexible Printed Wiring and Flexible Bonding Films

IPC-FC-241 Metal Clad Flexible Dielectrics for Use in Fabrication of Flexible Printed Wiring

IPC-FA-251 Assembly Guidelines for Single-Sided and Double-Sided Flexible Printed Circuits

IPC-TM-650 Test Methods Manual

Method 2.4.18.1 Tensile Strength and Elongation, In-House Plating

IPC-SM-782 Surface Mount Design and Land Pattern Standard

IPC-SM-840 Qualification and Performance for Permanent Solder Mask

IPC-2221 Generic Standard on Printed Board Design

IPC-4101 Specification for Base Materials for Rigid or Multilayer Printed Boards

3 GENERAL REQUIREMENTS

General requirements **shall** be in accordance with IPC-2221 and as stated in 3.1 through 3.4.2.

3.1 Design Modeling The design cycle should include full-sized three-dimensional modeling of the design to assure correct dimensioning and layout of flexible and rigid circuit areas (see Figure 3-1).

3.2 Design Layout The circuit layout depicts the physical size, flexible section, and location of all electronic and mechanical components, electrical schematic, and electrical requirements (e.g., impedance, amperage, voltage) to allow preparation of documentation and artwork.

3.2.1 Mechanical Layout Efficiency (Consider Final Panelization) Because flex designs may take a variety of shapes, it is recommended that the designer consider the use of folding to accomplish efficient panelization (see Figure 3-2). Layout efficiency is important because of the benefits in reduced processing cost and material usage. However, these savings can be offset by the labor required for the folding in the final assembly. Each fold will change the coverlay access opening orientation of the single-sided flexible circuit.

To achieve panel efficiency, consideration should be given to alternative interconnections such as discrete wiring. In many cases, it is possible to eliminate costly or lengthy flex extensions by the simple attachment of wiring or cable.

3.2.2 Fabrication Drawing Recommendations Separate views showing an installed flex configuration should be added to the fabrication drawing. The intent is to provide

1. IPC, 2215 Sanders Road, Northbrook, IL 60062, 847-509-9700, www.ipc.org

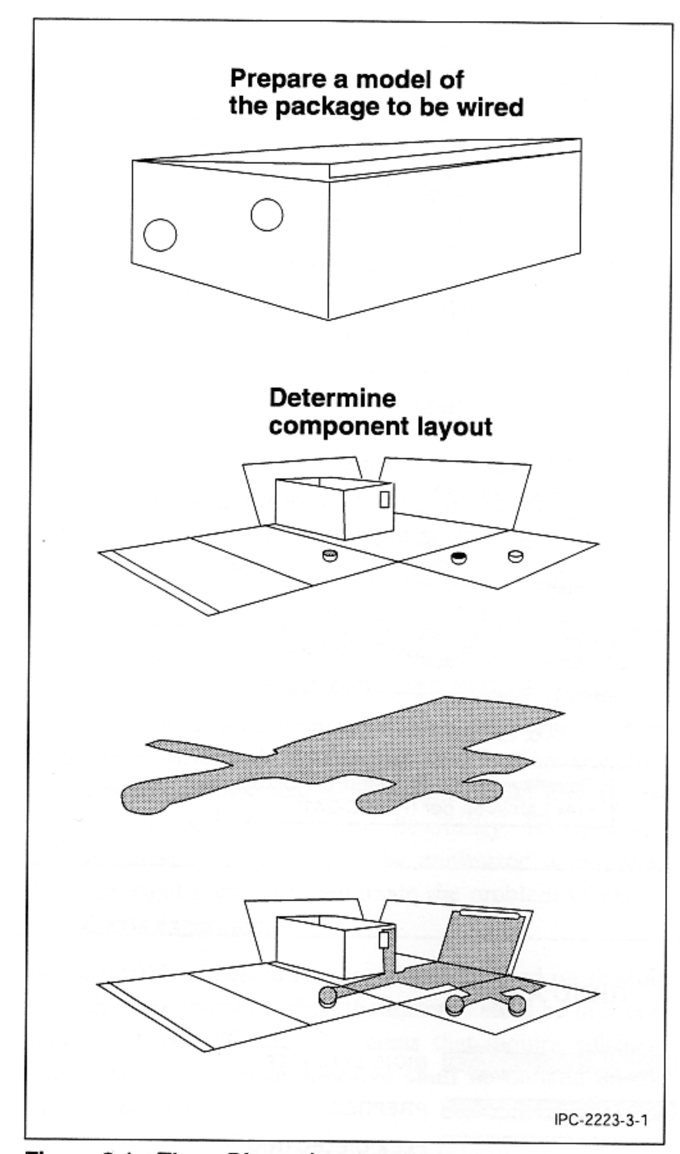


Figure 3-1 Three Dimensional Modeling

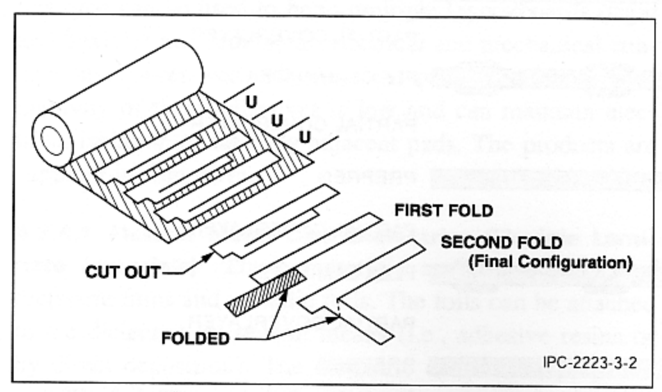


Figure 3-2 Final Panelization

the fabricator with the locations of the critical areas that are to be folded or flexed.

Drawings should contain a detailed list and description of the materials contained in the flex construction (e.g., materials stack-up, reinforced/stiffened areas, and critical thickness areas). A cross-sectional view is recommended. For critical thickness areas, the upper and lower ends of the thickness requirement should be defined.

3.3 Schematic Pin out considerations should be considered early in the design to avoid complicated and costly crossovers in the flex connectivity. Pin out considerations can reduce layers and therefore reduce cost.

3.4 Test Requirement Considerations Electrical testing for flex circuits may include connectors and components. Specifications need to be given to the fabricator as to the electrical requirements of the end product. Test requirement considerations **shall** be in accordance with IPC-2221.

3.4.1 Environmental The flex circuit should be tested in the end-use environment for design verification.

3.4.2 Mechanical/Flexural To evaluate flex life, it is recommended that flexural testing be done in a manner that reflects intended end product use.

4 MATERIALS

4.1 Material Selection Material type and construction is extremely important in designing flexible printed wiring. All materials **shall** be specified on the master drawing. For clarification, it is suggested that cross-sectional views be used to highlight material selection. Examples of this are shown in Figure 4-1 and Figure 4-2.

4.1.1 Material Options At the fabricator's option, flexible metal clad dielectrics and adhesive coated dielectric films may be manufactured using individual components per IPC-MF-150, IPC-FC-231, and IPC-FC-232. In addition, materials per IPC-FC-241 and IPC-FC-232 may be substituted where individual components are specified. These documents group materials into slash sheets that are generic in nature. This means that materials meeting the minimum requirements have widely different typical properties. It is important to research the various products to choose the one best meeting the design requirements. The attributes that should be considered are:

- Moisture absorption
- Fire retardancy
- Electrical properties
- Mechanical properties
- Thermal properties

The designer and fabricator should concurrently review material selection for cost, performance, and producibility.

4.2 Dielectric Materials (Including Prepreg and Adhesives)

4.2.1 Preimpregnated Bonding Material (Prepreg) Prepregs are used in the fabrication of rigid-flexible printed

Figure 4-1 Flexible Area Construction Examples

Figure 4-2 Unbonded Flex Area Construction of Rigid Flex

wiring. "No-flow" and "low-flow" types are typically used in the bonding of the rigid sections. The higher glass transition temperature (T_g) materials used in these prepregs offer higher operating temperatures and lower Z-axis expansion coefficient, which directly affect plated-through hole (PTH) reliability for <8 layers (see 5.2.2.2). Disadvantages of using these materials include lower dielectric strength and reduced flexibility.

When preimpregnated bonding materials are used on flexible and rigid-flex printed wiring, the material **shall** be in accordance with IPC-4101. The areas that require adhesive and those to be free of adhesive **shall** be defined on the master drawing.

4.2.2 Adhesives (Liquid) Adhesives such as epoxy, acrylic, or silicone can be used to provide strain relief at rigid-flex transition areas.

4.2.3 Flexible Adhesive Bonding Films (Cast Adhesive or Bondply) Flexible adhesive bonding films are typically used in bonding multiple flexible layers and attachments used for thermal management or structural support. These materials offer high bond strength to the flexible dielectric. These bonding films may be formulated from low T_g resins to enhance adhesion and flexibility. In rigid-flex designs, these materials should be minimized or removed from the rigid sections to eliminate the problem of excessive Z-axis expansion.

When flexible adhesive bonding films are used on flexible and rigid-flex printed wiring, the material **shall** be in accordance with IPC-FC-232. The areas that require adhesive and those to be free of adhesive **shall** be defined on the master drawing.

4.2.4 Conductive Anisotropic Adhesives Adhesives of this type can be used to bond multiple layers/boards (rigid and flexible) and provide an electrical and mechanical connection between vertically adjacent pads. The lateral conductivity of these adhesives is low and can maintain electrical isolation of laterally adjacent pads. The products are supplied in film form.

4.2.4.1 Flexible Metal Clad Dielectrics (Flexible Laminate Materials) These materials are combinations of dielectric films and metallic foils. The foils can be attached to the dielectric by several means (i.e., adhesive resins or by direct deposition). The dielectric can be cast onto the metallic foil. The cast dielectric laminates and direct deposition laminates are called adhesiveless. The traditional laminate configuration is constructed via the use of adhesive resin to bond the dielectric film to the metal foil. The T_g of these adhesive resins is usually lower than the dielectric film. High layer rigid-flex designs are currently employing adhesiveless laminates to minimize the impact of the low T_g adhesives (see Table 4-1).

Flexible metal clad laminates **shall** be in accordance with IPC-FC-241 or combinations of IPC-MF-150, IPC-FC-231, and IPC-FC-232.

4.2.4.2 Rigid Metal Clad Laminate Materials These laminate materials are combinations of metallic foils, resins, and woven or non-woven reinforcements. Resins used in these laminates offer a wide range of T_g (see IPC-2221).

4.2.5 Coverlayer The coverlayer is a combination of dielectric films and adhesives or a flexible dielectric coating. The coverlayer is used to insulate/isolate the conductor layers on the surface of flexible printed wiring. The coverlayer is constructed of materials that can be flexed or formed in the intended use. There are two types of coverlayers: coverfilms and covercoats.

4.2.5.1 Coverfilm The coverfilm is comprised of dielectric films and adhesives. For dynamic applications, it is important to balance the circuit and coverfilms.

4.2.5.2 Covercoat The covercoat is a coating that can be applied by dry film lamination, screening, spraying, or dipping/curtain coating. The resin can be formulated to be photoimageable. Intricate pad designs or tight pad spacing can be addressed/answered through the use of the photoimageable types.

Some covercoats may be in accordance with IPC-SM-840. Selection of material for flexibility is critical.

4.3 Conductive Materials (Surface Finishes) Conductive materials **shall** be in accordance with IPC-2221 and as stated in 4.3.1 through 4.3.7.

4.3.1 Electrolytic Copper Plating

4.3.1.1 Flex-to-Install Applications Copper elongation is recommended to be 12% or greater for low flex life applications such as PTHs and circuit flex-to-install. The copper elongation is to be measured in accordance with IPC-TM-650, Method 2.4.18.1.

4.3.1.2 Dynamic Flex Applications Copper elongation above 18% is recommended if used in high flex life cycle applications. The copper elongation is to be measured in accordance with IPC-TM-650, Method 2.4.18.1. Additional electrolytic plating on the surface of the base material is not recommended in flexible areas. This suggestion is based on the fact that flexibility is adversely affected by thickness of the conductor. For additional information, see 5.2.3.

4.3.1.3 Hole Plating Minimum average copper thickness for PTHs and vias are as shown in Table 4-2 and are the same for all classes. These are different from those in

Table 4-1 Characteristics of Flexible Dielectrics[1]

	POLYESTER (with Adhesive)	POLYIMIDE (with Adhesive)	POLYIMIDE (Adhesiveless)
MECHANICAL			
FLEXING (R ~ 2.0 mm)	FAIR	GOOD	EXCELLENT
THERMAL FORMING	YES	NO	NO
MODULUS	2800 MPa – 5500 MPa	2500 MPa	4000 MPa
TEAR STRENGTH	800 g	500 g	500 g
PEEL STRENGTH (AMBIENT)	1050 N/M	1750 N/M	1225 N/M
CHEMICAL/ENVIRONMENTAL			
CAUSTIC (>20%)	EXCELLENT	POOR	GOOD
UV	POOR-PET/FAIR-PEN	GOOD	EXCELLENT
UL RECOGNITION/MAXIMUM OPERATING TEMP.	85°C–160°C	85°C–160°C	105°C–200°C
FLAME RETARDANCY	VTM-0 WITH FR ADHESIVE	VTM-0 WITH FR ADHESIVE	VTM-0
ELECTRICAL			
DIELECTRIC CONSTANT (1 MHz)	3.4	3.5	3.3
DIELECTRIC STRENGTH	4-5 Kv/25 µm	3-5 Kv/25 µm	5 Kv/25 µm
INSULATION RESISTANCE	10^3 Ω–cm	10^3 Ω–cm	10^3 Ω–cm
THERMAL			
SOLDER PROCESSING	5 sec @ 246°C–260°C	5 sec @ 288°C (Predry Req.)	10 sec @ 288°C (No Predry)
ASSEMBLY			
THROUGH HOLE	LIMITED	EXCELLENT	EXCELLENT
SURFACE MOUNT (IR REFLOW)	PEN, YES PET, NO	GOOD TO EXCELLENT	EXCELLENT
WIRE BONDING	NO	SOME ADH. ARE OK	EXCELLENT
CHIP (DIRECT ATTACH)	POOR	FAIR TO EXCELLENT	EXCELLENT

[1] The stated values are typical and will vary among different material suppliers. Consult the laminate manufacturer utilized by the fabricator for specific values.

Table 4-2 Minimum Average Copper Thickness

	Type 2	Type 3	Type 4[1]	Type 3,4[2]
Overall Flex Thickness	≤0.2 mm	>0.2 mm	>0.2 mm	>0.75 mm
Minimum Average	12 µm	25 µm	25 µm	35 µm
Minimum Thin Area	10 µm	20 µm	20 µm	30 µm

1. Requirements for constructions with a ≤10% content of low T_g materials.
2. Requirements for constructions with a ≥10% content of low T_g materials.

IPC-2221 because of the unique geometry of double-sided and low T_g materials used in multilayer and rigid-flex. For etchback of holes, see 9.2.2.1.

4.3.1.4 Selective Plating Requirements When the design is to be used in an application where conductor thickness is critical, selective plating is suggested. When selective plating is required, a "pad only" (button plate) artwork may be generated by the fabricator. This artwork can be generated by a number of methods. Special precautions may be needed when using clipped or shaved pads. Care should be taken in the design to use the largest possible pad size.

It should be noted that some manufacturing processes will leave a small amount of plating on the surface of the conductors. Certain areas may not tolerate any plating; these areas **must** be stated on the master drawing (see Figure 4-3).

4.3.2 Nickel Plating Nickel plating over the flexible section is not recommended due to its brittle nature. Cracks in the barrier coating will propagate and cause failure in the copper conductor.

4.3.3 Tin-Lead Plating Unless otherwise specified, all tin-lead plating **shall** be fused.

Caution: Not all flex circuit base materials are capable of surviving the temperatures required for fusing. See Table 4-1 for material selection.

4.3.4 Solder Coating

Caution: Not all flex circuit-based materials are capable of surviving the temperatures required for solder coating. See Table 4-1 for material selection.

4.3.5 Other Metallic Coatings These requirements **shall** be specified on the master drawing.

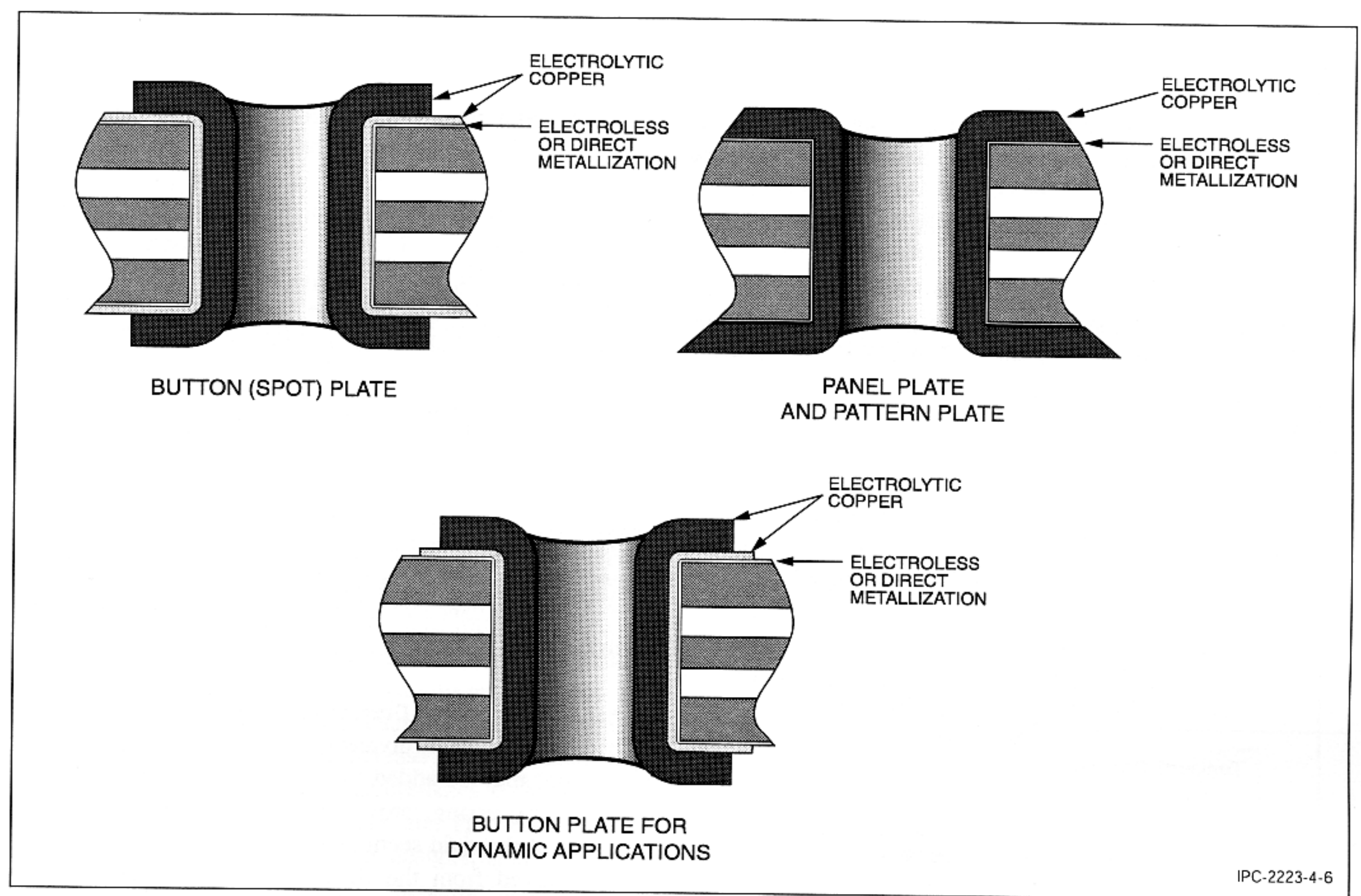


Figure 4-3 Selective Plating

4.3.6 Electronic Component Materials (Buried Resistors and Capacitors)

Caution: These materials should not be used in flex areas. They are primarily used in the rigid sections of rigid flex.

4.3.7 Conductive Coatings for Shielding Conductive inks such as silver, copper, or carbon-filled polymers can be applied to the surfaces of dielectric layers to provide shielding. These coatings should be specified on the master drawing. Connection to ground can be made by direct attachment through an opening in the dielectric.

4.4 Organic Protective Coatings Organic protective coatings should be in accordance with IPC-2221 and as stated in 4.4.1 and 4.4.2.

4.4.1 Solder Resist Though commonly used in rigid printed wiring, soldermask selection **must** be considered if used in any flex area. There are specific products that are designed to be used in a wide variety of flex applications.

4.4.2 Conformal Coating Conformal coatings should generally be omitted from flexible areas to avoid stiffening of the flex structure and debonding of the flex coating.

4.5 Marking and Legends It is good practice to avoid marking in the flexible printed circuit dynamic area. Marking **shall** be defined on the procurement documentation or master drawing.

5 MECHANICAL AND PHYSICAL PROPERTIES

Mechanical and physical properties **shall** be in accordance with IPC-2221 and as stated in 5.1 through 5.4.1.

5.1 Fabrication Requirements See 3.1 through 3.2.1 for design modeling and mechanical layout efficiency.

5.1.1 Bare Board Fabrication The manufacturer should be consulted on the panel sizes available and useable area on the panel. Typical panel sizes are 46 cm x 61 cm or 31 cm x 46 cm.

5.1.2 Roll to Roll Fabrication The manufacturer should be consulted on web width and roll length, as the variations are determined by equipment and material thickness used in the application.

5.1.3 Borders and Spacing The manufacturer should be consulted on useable area and spacing, as this depends on the material and processing parameters.

5.2 Product/Board Configuration

Note: The board standardization figure in IPC-2221 may not apply to flexible or rigid-flex printed wiring due to unique profiles. Nesting techniques vary based on profiles and the need to maximize material utilization.

5.2.1 Circuit Profile (Outline) The exterior outline of the proposed circuit should not waste raw material. If the circuit has peninsulas or fingers extending in many directions, material costs may be higher than necessary. Folds in the circuit may permit extensions to be manufactured near the main body of the circuit and provide a more compact, rectangular circuit outline for processing.

5.2.1.1 Minimum Radius (Flexible Sections) The minimum radius on inside corners of the part profile should be 1.6 mm; however, larger radii will make a more reliable part and be more resistant to tearing (see Figure 5-1 and Figure 5-2).

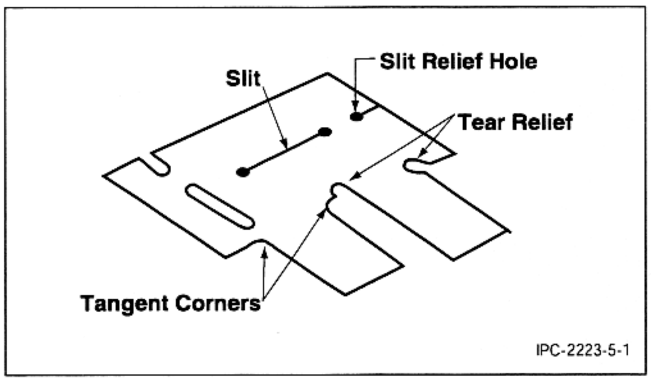


Figure 5-1 Special Flexible Printed Wiring Features

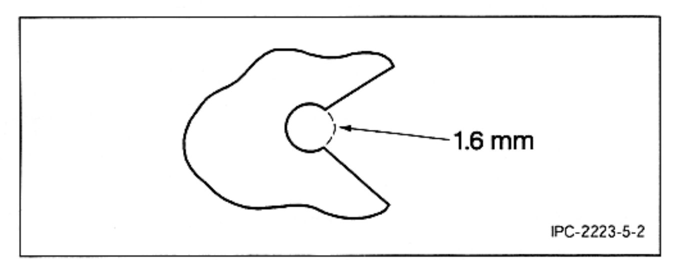


Figure 5-2 Cutout with a Drilled Hole

5.2.1.2 Hole to Edge Distance (Flexible sections) The minimum distance between exterior edges and the edges of interior holes and cutouts should not be less than 1.3 mm. A larger distance will make a more reliable part.

5.2.1.3 Hole to Edge Distance (Rigid Section) Exterior edges and the edges of clearance holes and PTHs should not be less than 1.9 mm apart. A larger distance will make a more reliable part. This minimum distance also applies to the rigid to flex transition edge (zone).

5.2.1.4 Counterbores and Countersinks Counterbores and countersinks should be avoided due to the complexities of accurate machining in rigid flex structures with practical thickness tolerances.

5.2.1.5 Variable Thicknesses The rigid laminated sections of a multilayer flex and rigid flex should be the same thickness to accommodate the processing of PTHs. Sequential lamination or varied thickness adds to processing and cost.

5.2.2 Rigid Area Considerations (Multilayer and Rigid Flex)

5.2.2.1 Bow and Twist Due to the nature of flexible and rigid material combinations, special construction, tooling, and/or fixturing may be required to meet surface mount requirements. When possible, the bow and twist requirements should apply to the rigidified sections.

5.2.2.2 PTH Reliability To minimize the amount of Z-axis expansion, the percentage of low T_g (e.g., acrylic) adhesive in the rigid section should be kept to a minimum. This can be accomplished with the use of adhesiveless-based materials and partial coverlayers of the flexible layers (see Figure 4-2). It is recommended that prepreg materials in accordance with IPC-4101 be used as bonding films in the rigid section.

5.2.2.3 Additional Dielectric Material Additional dielectric material of equivalent characteristics (i.e., pouch or cocoon) may be added to the rigid section construction to aid in processing, provided the overall thickness requirements of the rigid sections are not violated. Pouch material is removed from the flexible areas after processing and should be kept to a minimum extending from the rigid area after removal (see Figure 4-2).

5.2.3 Flexible Areas/Flexible Circuits

5.2.3.1 Flexible Area Considerations Factors to consider in determining the total number of layers required (along with other interrelated considerations) are:

- Quantity of signal traces required across the flexible portion(s)
- Line widths required for current-carrying capacity
- Spacing required for voltage isolation
- EMI shielding
- Impedance
- Voltage drop requirements
- Mechanically defined "real-estate" for routing the traces (i.e., the width of the flexible portions)

As an example, a relatively low quantity of required traces with low current carrying capacities could be fabricated with a single-sided layer with 34 µm copper (0.036 mm thick) traces on an adhesiveless copper-clad laminate with a 0.050 mm base thickness using a 0.025 mm coverlayer film with 0.025 mm adhesive thickness for a total thickness of 0.136 mm. However, if the current carrying capacity required across the traces dictates 69 µm copper (0.071 mm thickness), the adhesive thickness of the coverlayer would

have to also be increased to 0.050 mm (to properly encapsulate the traces), and the total thickness would increase to 0.198 mm.

Note: The minimum coverlayer adhesive thickness should be 0.025 mm per 34 µm of copper thickness.

Another factor, along with the number of conductive layers, at this point should be other mechanical considerations. If there were a concern for mechanical strength, the thickness of the coverlayer film may have to be increased to 0.050 mm or more, independent of the adhesive thickness. This consideration would increase the thickness of the single-sided layer in the above example to 0.145 mm for the 34 µm copper and to 0.225 mm for the 69 µm copper.

Caution Note: In the single-sided layer, it has been shown how one aspect of the conductors, namely the current carrying capacity, can have a significant effect on physical properties such as thickness. There are other electromagnetic characteristics of the conductors, such as capacitance, inductance, resistance, etc., that can likewise affect the design. These characteristics can cause effects such as cross talk. Greater separation or an increase in layer count may be required due to electrical characteristics such as power and signal separation, high voltage, etc. The reader should be aware of these factors in their designs, since they can also affect flexibility and stiffness characteristics.

If the number of conductor traces required becomes too high for the allowed width of the flexible portion(s), then a double-sided layer would be required. This would also occur if, for example, the impedance needs dictate an imbedded microstrip configuration wherein one of the conductive layers is a reference plane. The thickness of the double-sided layer will be subject to the same type of considerations used for the single-sided layer above.

Continuing the above line of reasoning, a multilayer configuration would be needed if the impedance requirements dictate a stripline configuration or if EMI shielding is required on both the top and bottom sides of a sensitive conductor, for example. Dielectric spacing calculations between the conductor and the adjacent reference planes of a stripline configuration **must** take into account the dielectric constant (Er) of the material used for the flexible portion(s), line width, and copper thicknesses needed to achieve desired impedances. The final dielectric thickness between conductive layers may be less than originally calculated due to construction and process parameters.

While higher layer counts in the flexible areas (four conductive layers and above) can be produced, they are not recommended due to larger bend radii required for the thickness and induced stresses to the material(s). Should higher layer count designs be required, it is suggested that mechanical testing be performed. It is to be noted that as we progress from a single-sided layer to a multilayer, there is a decrease in flexibility. For dynamic flex applications (Installation Use B), a double-sided layer would be the thickest construction recommended (see Figure 4-1).

5.2.3.2 Flexible Area Considerations - Multi Layer and Rigid Flex Multilayer flexible printed wiring is not very flexible. If flexibility is required, it can be achieved by not laminating together specific sections of the cable, as shown in Figure 4-2. This type of design should be used in multilayer constructions containing more than four layers of flexible material.

Caution Note: Consideration **shall** be given to the flexible area between the bonded areas in regards to the number and construction of layers and length of the flexible area. The shorter the distance between the bonded areas, and as the bend angle increases, the corresponding increase in compression of the innermost layer may cause buckling. These areas of buckling may cause small isolated areas of reduced radii (see Figure 5-3).

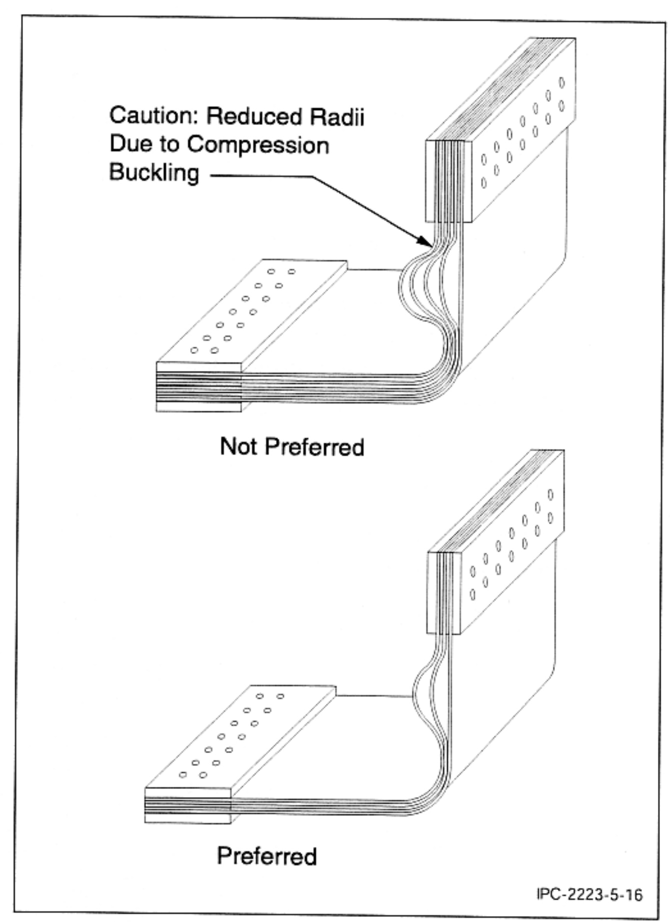


Figure 5-3 Reduced Bend Radii

5.2.3.3 Bend Area Conductor Considerations The flexible printed wiring **shall** not be flexed or formed in an area where there is discontinuity in the covercoat, termination of plating or potting, or any other stress-concentrating feature.

For maximum dynamic flex life (Use B) and maximum reliability for flex-to-install (Use A), conductors in the bend area (see Figure 5-4) should adhere to the following considerations:

- Perpendicular to the bend
- Evenly spaced across the bend area
- Maximized across the bend area
- Without additional plated metals
- Uniform in width
- Conductors in double-sided circuits should not be placed directly over each other, which produces an "I" beam effect. This condition may be necessary due to electrical considerations; however, mechanical installation requirements must be considered (see Figure 5-5).
- The number of layers in a bend area should be kept to a minimum.
- Vias and PTHs in bend areas should be avoided.
- The neutral axis, where possible, should be located at the center of the conductor. A balanced construction can be achieved by using materials of equivalent modulus values and thickness on each side of the conductor. This is critical in dynamic flexible printed applications. Several design techniques are popular for approximating this condition, such as using coverlays and interdigitizing conductors front to back (see Figure 5-5 and Figure 5-6).

5.2.3.4 Calculation of Bend Radius

5.2.3.4.1 Estimation of Minimum Bending for Single-Sided Circuits with Coverlay It is possible to estimate the minimum bending radius for single-sided flexible circuits (see Figure 5-7), where:

R = minimum bending radius in µm
c = copper thickness in µm

D = dielectric thickness in µm
E_B = amount of copper deformation in %

d = flexible clad dielectric thickness

$$R = (c/2)[(100-E_B)/E_B] - D \quad \text{[Equation 1]}$$

By setting the deformation of copper allowed, it is possible to determine the minimum bend radius. For one time crease designs, use the ultimate elongation at break (for rolled annealled copper, this value is 16%). For flex-to-install designs, use the IPC-MF-150 minimum elongation (for rolled annealled copper, this value is 10%). For dynamic flex designs on copper, use 0.3%.

Example:

50 µm Polyimide, 25 µm adhesive, 35 µm copper

Therefore, D = 75 µm, c = 35 µm

Total thickness of flexible circuit T = 185 µm

One time crease, use 16%	R = 16.9 µm, or a R/T = 0.09
Flex-to-install, use 10%	R = 0.08 mm, or a R/T = 0.45
Dynamic flex, use 0.3%	R = 5.74 mm, or a R/T = 31

5.2.3.4.2 Estimation of Minimum Bending Radius for Double-Sided Circuits with Coverlay

Base Material:	50 µm polyimide; 2 x 25 µm adhesive 2 x 35 µm cu d = 100 µm; c = 35 µm
Coverlays:	25 µm polyimide; 50 µm adhesive D = 75µm
Total Thickness:	T = 2D + d + 2c = 320 µm

$$R = (d/2 + c) \times [(100-E_B)/E_B] - D \quad \text{[Equation 2]}$$

From Equation 2:

Dynamic Flex	E_B = 0.3%	R = 28.17 mm	R/T = 88
Flex-to-Install	E_B = 10%	R = 0.690 mm	R/T = 2.15
One Time Crease	E_B = 16%	R = 0.371 mm	R/T = 1.16

5.2.4 Preforming Bends It is good design practice to minimize the amount of preforming and bending due to the memory characteristics of some flexible dielectric materials.

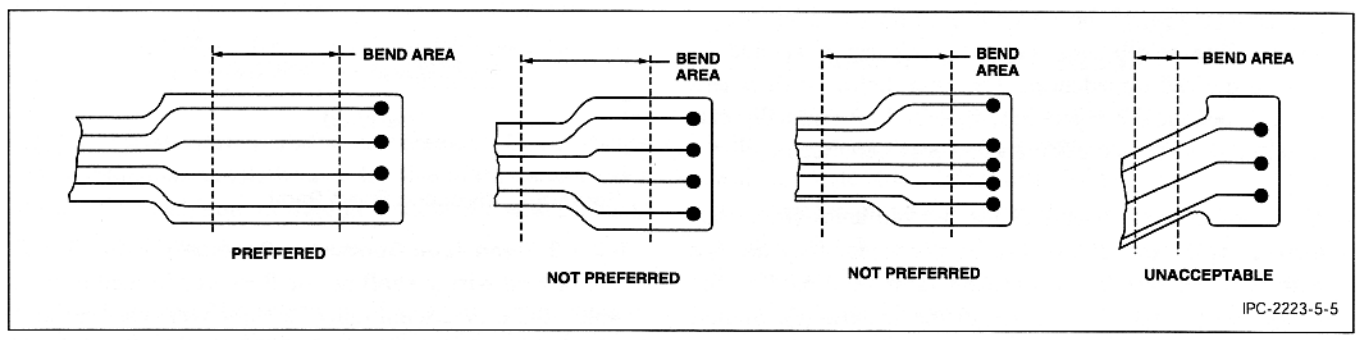


Figure 5-4 Conductors in Bend Areas

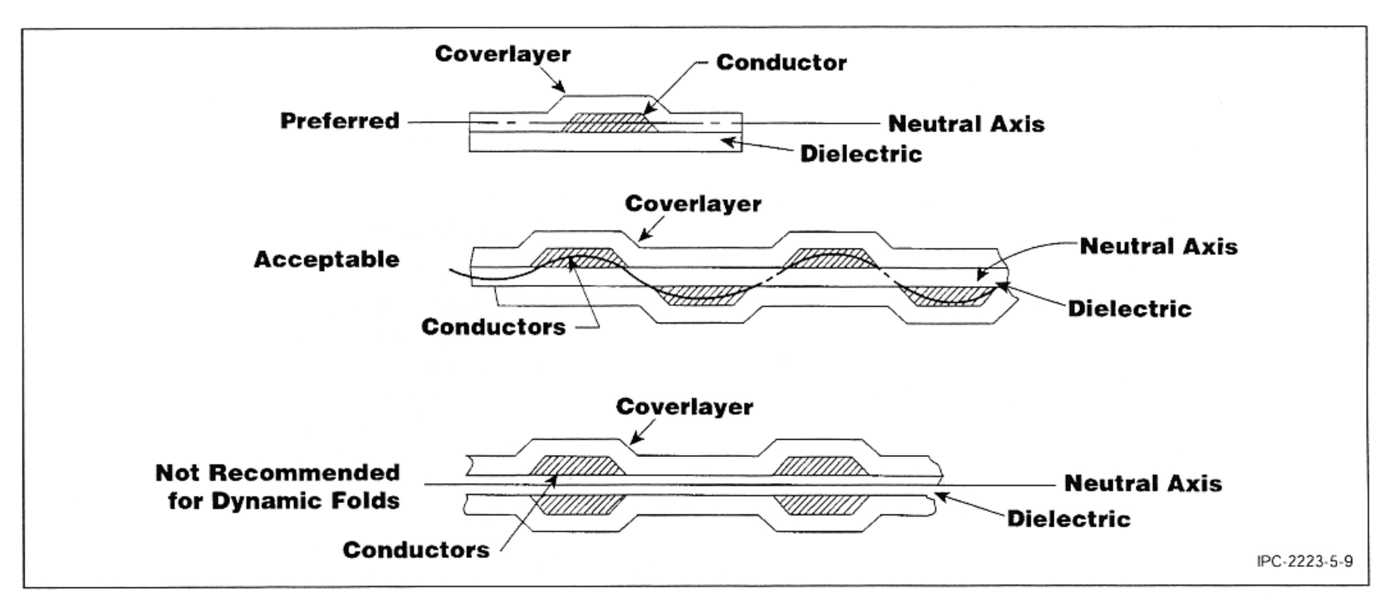


Figure 5-5 Bend/Crease Areas Center Lines

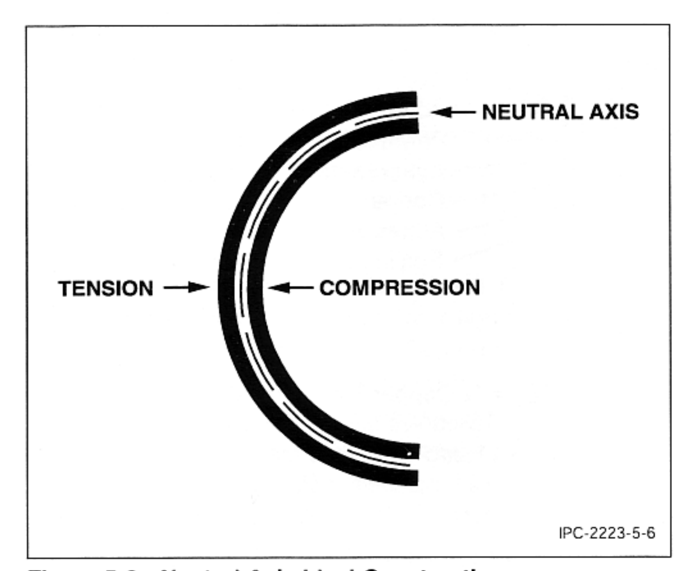


Figure 5-6 Neutral Axis Ideal Construction

Bends or creases can be formed in flexible circuits. The principle concern in low flex and flex-to-install applications is high ductility of the copper. The formability and reliability of a circuit is dependent upon the thickness and available ductility of the copper foil, as well as the substrate material and adhesive system employed. Three methods of permanently forming circuits are employed: cold forming, thermal forming, and forming through reliance on the properties of the copper used either as a conductor or simply left unetched as a reinforcement. Thermal forming is generally only used with substrate/adhesive combinations from the same materials family (i.e., polyimide, polyester, etc.). Thermal forming of the polyimide materials is highly dependent on adhesive, copper, and dielectric thickness. Preforming requires expensive manufacturing, tooling, and shipping containers. When preforming is required, it should be performed just prior to installation in the final unit. The addition of a strain-relief bar or some other means of support is required if bending occurs close to a solder joint, PTH, or rigid to flex transition point. Bend and/or crease line drawing requirements are as follows:

- Bend/crease lines should be shown on fabrication and assembly drawings.
- Bend/crease line dimensions **shall** be depicted as a reference. The assembly drawing should show the finished folded configuration as a reference only.
- Bend/crease lines should be depicted as center lines and described as "bend line corresponds to center of arc formed when folded."

5.2.4.1 Bends or Folds (Greater than 90°) The baring of terminal pads on both sides of a flexible cable is difficult to achieve. Sometimes it is more practical to fold it, as shown in Figure 5-8. Folds should be kept uniform and designed to follow the surfaces of the package. These types of folds are designed for single-sided flexes. Terminal rows on single-sided flexible printed wiring can be made to appear double-sided by folding the circuit. This will be less expensive than the use of double-sided PTH flexes in almost every design. A small rod or wire mandrel may be employed inside the fold or folds to control the bend radius and prevent conductor cracking. The folded portion of the circuit may be secured with an adhesive.

A crease, if required, **shall** be formed only one time. Once formed, it **shall** not be opened again.

Keep in mind the bend allowances required in tight corners. Do not let regular folds "bind" against parts since such binding areas can be points of weakness under vibration (see Figure 5-8 and Figure 5-9).

5.2.4.2 Flex-to-Install Radii The bend radii should be kept as large as possible. The suggested minimum bend

Polyimide Coverlayer (D)
Adhesive layer
Copper (c)
Adhesive layer (D)
Polyimide base material
Mandrel radius **(R)**

E_B = % Copper Elongation desired
- Rolled Annealed max. ≤16%
- Electro-deposited max. ≤11%
- Flex to install applications ≈3%
- Dynamic flex applications ≈0.3%
- Disk drive applications ≈0.1%

R = radius of fold
c = copper thickness
D = dielectric thickness with adhesive

Single-sided flexible section

Polyimide coverlayer (D)
Adhesive layer
Copper (c)
Adhesive layer (d)
Polyimide base material
Adhesive layer
Copper
Adhesive layer (D)
Polyimide coverlayer

E_B = % Copper Elongation desired
- Rolled Annealed max. ≤16%
- Electro-deposited max. ≤11%
- Flex to install applications ≈3%
- Dynamic flex applications ≈0.3%

R = radius of fold
c = copper thickness
D = dielectric thickness with adhesive
d = flexible clad dielectrics thickness

Double-sided flexible section

IPC-2223-5-7

Figure 5-7 Stresses on Layers During Folding

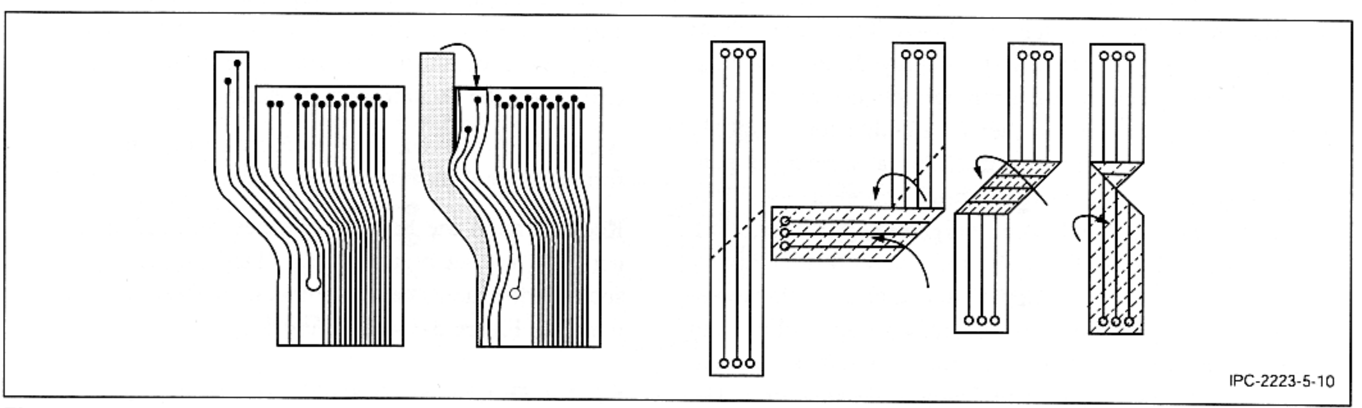


Figure 5-8 Fold Back

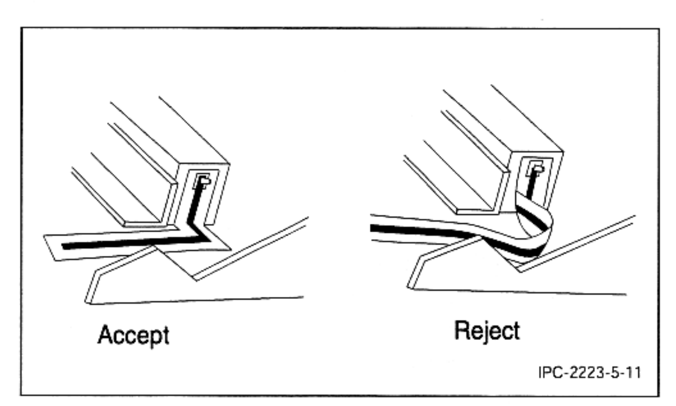


Figure 5-9 Irregular Folds

radius should be 10 times the overall thickness of the completed flexible printed wiring. It is preferable that flexible printed wiring be allowed to follow its own natural bend. Bends greater than 90° and containing a small radius should be avoided.

5.2.4.3 Bonded Multilayer Bending Bonded multilayer flexible printed wiring is not as flexible as single or double-sided flexible printed wiring. If flexibility is required, it can be achieved by not laminating specific sections of the cable together. This type of design should be used in multilayer constructions containing more than four layers of flexible material.

The maximum number of copper layers in a multilayer configuration should be limited to four layers if bending of the parts is required (see 5.2.3.1). However, once bent, the flexible multilayer **shall** not be flattened or re-bent on the same axis. Since all of the materials are inherently less stable than those used to produce rigid multilayer boards (the multilayer flexible printed wiring depends on the conductors being able to bend freely without elongation), a third or fourth layer would bring the conductor too far from the neutral bend axis to prevent the copper from stretching.

5.2.5 Differential Lengths

5.2.5.1 Differential Lengths (Flexible Printed Wiring) When an assembly application requires that two or more flexible printed boards be selectively bonded or fastened together at the same common termination points at either end and are bent at least 90°, but without forming an "S" shape in a single flex location, then differential board lengths should be incorporated in the design of the part (see Figure 5-10 and 5-11). These differential board lengths are calculated for the bent portion of successive boards. The following is a general expression for the differential length calculation.

The calculation assumes the following:

- Only a single bend radius per flexible printed board is considered.

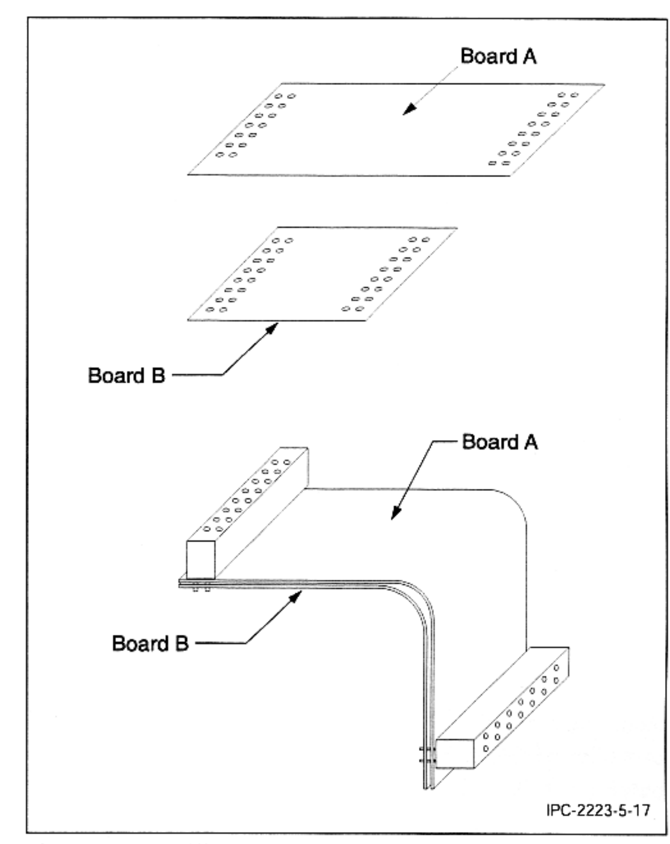


Figure 5-10 Differential Board Lengths

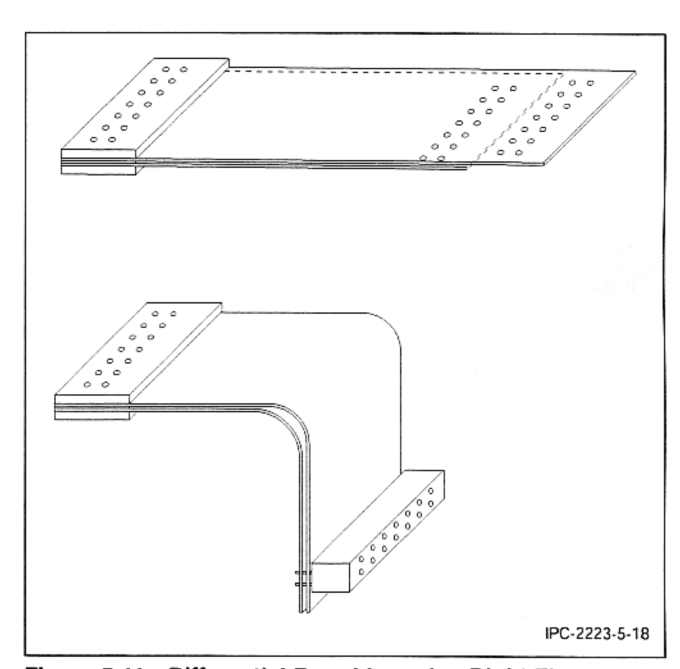


Figure 5-11 Differential Board Lengths, Rigid-Flex

- The radius is assumed to be circular (i.e., a portion of the circumference).
- All flexible layers are assumed to be equal in thickness and separated by an adhesive layer of relatively small, uniform thickness.
- The effective bend radius of each layer is its neutral axis.

- The radius dimension should be to the worst case location tolerance (longest distance (r_0)) of the two end points.
- Staggered lengths are sometimes used to terminate multi-layer flexible printed wiring to a connector.

5.2.5.1.1 Calculation The basic bend length or portion of a circumference is given by:

$$L_0 = 2\Pi r_0 \, (\varnothing \div 360) \qquad \text{[Equation 3]}$$

where: L_0 = length
r_0 = basic inner bend radius to the first flex layer
$\varnothing$ = the angle, in degrees, through which the flex is bent

therefore, $L_0 = 2(3.1416)\,(r_0)\,(\varnothing \div 360) = 0.0175\, r_0 \varnothing$

Therefore, the effective bend radius of the first flex layer is:

$$r_1 = r_0 + T_L/2 \qquad \text{[Equation 4]}$$

where: T_L = total thickness of the flex layer, including covercoat. Thus the effective bend radius is the distance from the center of the curvature to the neutral axis of the flex layer. The length of the curved portion of the first flexible layer is:

$$L_1 = 0.0175 r_1 \varnothing$$

As successive layers are added to the first layer, the effective radius of the n^{TH} layer is:

$$r_n = r_1 + [(T_L + G)\,(n\text{-}1)] \qquad \text{[Equation 5]}$$

where: G = thickness of the adhesive joint
n = number of layers

Note: G is only used if the layers are to be laminated together. If the layers are to remain separated, then G = 0 and disappears.

The length of the curved portion of the n_{TH} layer is:

$$L_n = 0.0175 r_n \varnothing \text{ or}$$
$$L_n = 0.0175\varnothing\, [r_1 + (T_L + G)\,(n\text{-}1)] \qquad \text{[Equation 6]}$$

The differential length of interest is the difference in the curved length of the first flexible tape and the curved length of any successive layers:

$$\Delta L = L_n - L_1$$

From the above,

$$\Delta L = 0.0175\varnothing\, (r_n - r_1)$$
$$= 0.0175\varnothing\, [r_1 + (T_L + G)\,(n\text{-}1)] - 0.0175\varnothing\, [r_1]$$

Combining the above equation gives:

$$\Delta L = 0.0175\varnothing\, [(T_L + G)\,(n\text{-}1)] \qquad \text{[Equation 7]}$$

Note: The expression for the bend radius (r) has dropped out of Equation 7 and ΔL is governed to only be the degrees of the bend, the number of layers, the thickness of the flexible layers, and the associated adhesive bond joint thickness.

For example, if:

ΔL = increase in length per layer per bend
$\varnothing$ = 90° = angle of bend in degrees
T_L = 0.381 = thickness of layer in mm
G = 0.025 = thickness of adhesive per layer in mm
n = 4 = number of layers

Then, from Equation 7, the differential lengths required are as follows:

n	ΔL
1.	0.0175 (90) (0.381 + 0.025) (1-1) = 0 mm
2.	0.0175 (90) (0.381 + 0.025) (2-1) = 0.6394 mm
3.	0.0175 (90) (0.381 + 0.025) (3-1) = 1.278 mm
4.	0.0175 (90) (0.381 + 0.025) (4-1) = 1.918 mm

5.2.5.2 Differential Lengths (Multilayer and Rigid Flex) The bookbinder design of an unbonded flex area can be used in regions where a sharp bend (radius to thickness ratios <6) is required. This technique uses progressive lengths in the flex area (see Figure 5-12) and is costly to manufacture because of tooling complexity, processing difficulties, and reduced yields. Calculations used are the same as those in 5.2.5.1.

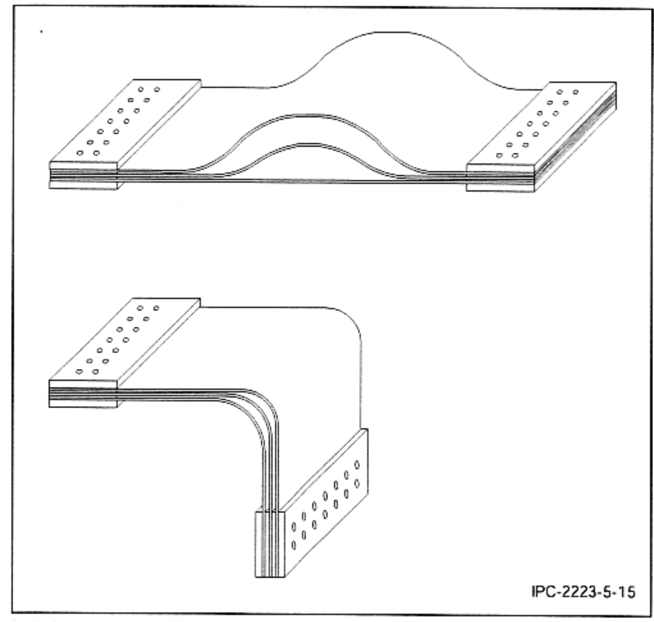


Figure 5-12 Bookbinder

5.2.6 Shielding A conductive layer may be added to provide shielding effectiveness. This layer may be a metal particle-filled resin applied in a thin coat over a signal layer or a metal foil. The shield **shall** be considered a layer in neutral axis calculations. To improve flexibility, the shield

layer **shall** be kept to a minimum thickness. To further increase flexibility and increase adhesion, the shield layer may be latticed or crosshatched as electrical characteristics allow.

5.2.7 Ground/Power Plane When the ground or power is a separate layer or a large trace, it is advisable to distribute, break up, or balance the conductor widths through the bend area. To improve flexibility and increase adhesion, the plane layer may be latticed or crosshatched as electrical characteristics allow.

5.2.8 Strain Relief Slits and slots should terminate in a 1.5 mm diameter or larger hole, as shown in Figure 5-13. This situation arises when adjacent portions of the flexible printed wiring must move separately.

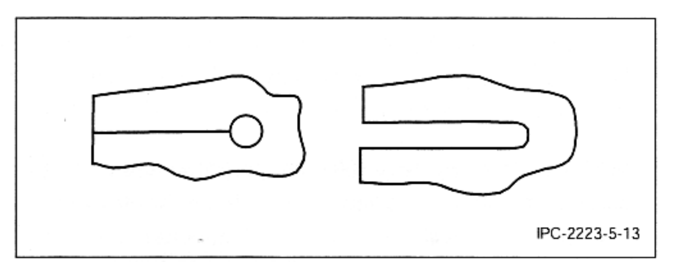


Figure 5-13 Slits and Slots

5.2.9 Adhesive Fillets (Strain Reliefs) When adhesive fillets (strain reliefs) are used at the transition of the flexible and rigid portions of Type 4 printed wiring, or at the transition of Type 1, Type 2, and Type 3 circuits with partial stiffeners, the fillet requirement should be defined on the master drawing. The materials can be flexible epoxies, acrylics, RTV silicones, polysulfides, etc. The fillet dimension is typically 1 mm to 2.5 mm from the rigid to flex interface. A large tolerance is required due to the flow characteristics of these materials. This is recommended to reduce stress at the interface when formed.

5.2.10 Stiffeners and Heat Sinks When stiffener, heat sinking materials, or other attachments are required, material types (metallic and nonmetallic), size, thicknesses, and adhesive type **shall** be specified on the master drawing. Size and registration of access holes in the stiffener to termination holes in the flexible and rigid-flex printed-wiring **shall** be defined on the master drawing. Access holes in stiffeners and heat sinks to expose lands should be at least 0.25 mm larger in diameter than the land to allow for registration tolerances and adhesive squeeze-out. The edge of the stiffener next to the flexible portion of the circuit should be radiused, chamfered, or adhesive filleted to prevent damage to the flexible printed wiring.

5.3 Assembly Requirements Assembly requirements **shall** be in accordance with IPC-2221 and as stated in 5.3.1 and 5.3.2.

5.3.1 Mechanical Considerations Specific mechanical characteristic data should be obtained from IPC-FC-231, IPC-FC-232, IPC-FC-241, and the material manufacturer's data sheets.

5.3.2 Palletized Flexible and Rigid Flex Printed Wiring Flexible printed wiring may be partially routed with breakaway tabs and maintained in the panel to facilitate subsequent handling, assembly, and test procedures.

5.3.3 Moisture If the dielectric is not free of moisture, soldering temperature may cause trapped moisture to boil. Depending on the amount of moisture present when the flexible printed wiring reaches the soldering temperature, delamination may be violent, literally blowing the circuit apart, or mild, producing small blisters, which could grow into serious delaminations.

For more information, see IPC-FA-251.

5.3.4 Infrared Preheats and Reflow Polyimide films absorb IR energy quickly because of their color. They must be monitored carefully when using IR preheats or reflow to prevent excessive heat buildup within the flexible printed wiring.

5.3.5 Adhesive T_g Flexible printed wiring can contain adhesives with very low T_g. For this reason, all thermal excursions during assembly should be monitored to prevent damage such as delamination and blistering. Preheat and soldering dwell times should be kept to a minimum. Special heatsinks or heat shields may be required.

5.4 Dimensioning Systems Dimensioning systems should be in accordance with IPC-2221 and as stated in 5.4.1.

5.4.1 Datum Features Datum feature or origins should be designated in each rigidized or localized termination area. The dimensions referenced to a datum for hole patterns should not include a flexible section.

The use of multiple datums, as shown in Figure 5-14, is a common technique for reducing or eliminating the effects of variable shrinkage or process distortion in flexible materials. One of the primary values of flexible printed wiring is its ability to be formed into three dimensions. With this flexibility comes inherent dimensional instability relative to hardboards or solid metal objects. Providing more positional tolerance from datum to datum, while specifying a tighter tolerance within each datum or termination area, allows for ease of manufacturing of the circuit without compromising subsequent installation of components in each termination area.

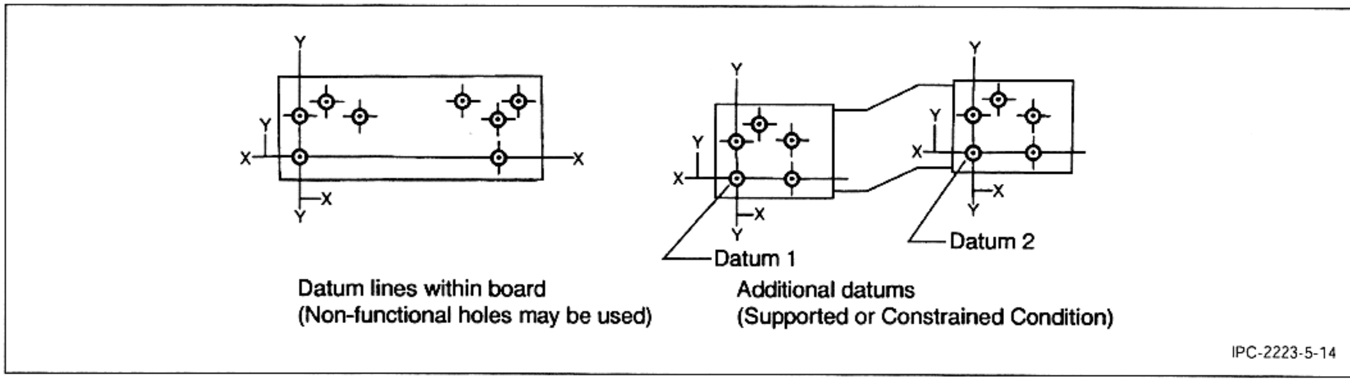


Figure 5-14 Establishing Datums

In assembly, the variable distance between datums can easily be accommodated by designing a small amount of slack or a service loop between the termination areas. For this reason, there is no loss of dimensional control or inferior quality associated with multiple datums.

6 ELECTRICAL PROPERTIES

Electrical properties **shall** be in accordance with IPC-2221 and as stated in 6.1 and 6.2. The tables and charts contained in IPC-2221 should be considered as guidelines only.

6.1 Electrical Considerations Continuous improvements in material and circuit technologies may allow reduced electrical clearances and/or greater current capacity. Specific electrical characteristic data should be obtained from the material manufacturer's data sheets.

6.2 Impedance and Capacitance Control When unbonded flexible impedance controlled layers are used in rigid-flex designs, impedance values may be compromised due to uncontrolled spacing between flex layers (see IPC-2221).

7 THERMAL MANAGEMENT

Thermal management **shall** be in accordance with IPC-2221 and as stated in 7.1.

Note: Thin dielectric materials have high electrical and low thermal resistance. The thermal characteristics of the specific flex dielectric should be reviewed for thermal transfer, T_g, and temperature index.

7.1 Heat Dissipation Considerations The use of metallic stiffeners in flexible printed wiring can allow forming, thermal management, and stiffening characteristics (see IPC-2221).

8 COMPONENT AND ASSEMBLY ISSUES

Because of the wide range of T_g material, component assembly methods and techniques should be reviewed.

8.1 General Placement Requirements General placement requirements **shall** be in accordance with IPC-2221 and as stated in 8.2 through 8.7.

The use of visual fiducials in the placement of parts on the flex should be placed in each rigidized or localized component mounting area.

8.2 Standard Surface Mount Requirements Standard surface mount requirements **shall** be in accordance with IPC-2221 and as stated in 8.3.

8.3 Lands for Surface Mounting Surface mount should only be used in rigid portions of rigid flex circuits or non-bending areas of flexible circuits. The selection of the design and positioning of the land geometry in relation to the part may significantly impact the solder joint. The designer **must** understand the capabilities and limitations of the manufacturing and assembly operations (see IPC-SM-782).

Special consideration should be taken for coverlay opening to compensate for adhesive squeeze-out over surface mount lands.

8.4 Constraints on Mounting to Flexible Sections Designs **shall** not place a component in an area of continuous flexing or in an area flexed, folded, or bent to install. Leads mounted through flexible material **shall** be fully clinched. If unclinched leads are required, supporting hardware, encapsulants, or stiffeners **shall** be designed as a part of the flexible printed wiring to ensure that no flexure-related stresses are exerted on the solder joints.

8.5 Interfacial Connections Interfacial connections on Type 2 flexible printed wiring **shall** be made by the use of clinched wires or PTHs. Interfacial connections on Type 3 and Type 4 flexible and rigid-flex printed wiring **shall** be made with PTHs only. Standoff terminals, eyelets, rivets, or pins **shall** not be used to provide interfacial connections.

Clinched wires used as interfacial connections are considered to be part of the assembly and **shall** be identified on the flexible printed wiring assembly drawing.

8.6 Offset Lands Lands, when used in conjunction with clinched leads, may be located adjacent to (not surrounding) the lead termination hole. The land **shall** be of a sufficient distance from the hole to allow clipping of the part lead prior to unsoldering the part lead from the land.

8.7 Terminal or Eyelet Terminals and eyelets can be used as a mechanical electrical connection where soldering can not be used or mechanical staking is needed.

9 HOLES/INTERCONNECTIONS

Holes and interconnections **shall** be in accordance with IPC-2221 and as stated in 9.1 through 9.2.3.

Note: For more information on materials and construction, see 5.2.2.2.

9.1 General Requirements for Lands with Holes General requirements for lands with holes **shall** be according to IPC-2221 and as stated in 9.1.1 through 9.2.2.1.

9.1.1 Land Requirements Land requirements **shall** be in accordance with IPC-2221. Filleting of line to pad transition is recommended in flexible printed wiring to reduce stress conditions and increase manufacturing tolerances (see Figure 9-1).

9.1.2 Annular Ring Requirements Annular ring requirements **shall** be in accordance with IPC-2221.

9.1.3 Eyelet or Standoff Land Area Considerations When eyelets or standoff terminals are used, the lands on Type 1, Type 2, and external layers of Type 3 and Type 4 **shall** be designed so as to have a minimum diameter of at least 0.5 mm greater than the maximum diameter of the projection of the flange, eyelets, or standoff terminals.

9.1.4 Land Size for Non-Plated Component Holes The pad size to be used **shall** be the largest possible pad that will meet the spacing requirements (see Table 9-1).

Table 9-1 Minimum Standard Fabrication Allowance for Interconnection Lands

Level A	Level B	Level C
0.5 mm	0.4 mm	0.3 mm

1. For copper weights >34 µm, add 0.05 mm minimum to the fabrication allowance for each addition µm of copper used.
2. For more than eight layers, add 0.05 mm.
3. See IPC-2221 for definitions of Level A, Level B, and Level C.

9.1.5 Land Size for Plated-Through Component Holes The pads size to be used **shall** be the largest possible land that will meet the spacing requirements. The land size used on the surface layers should be larger than the internal layers, if possible. The minimum standard fabrication allowance **shall** be per Table 9-1.

9.1.6 Thermal Relief in Conductor Planes When a ground and/or power plane is used, the thermal relief pad **shall** meet the requirements of IPC-2221.

9.1.7 Surface Mount Components See IPC-SM-782 for surface mount land pattern designs.

9.1.8 Nonfunctional Lands When pads are close together without enough room to put a conductor between them, it is allowable to remove unused pads on either side of the conductor. The minimum spacing between the conductor and the hole **shall** be 0.35 mm minimum plus the minimum spacing required.

If no connection to a plane is made, then the pad may be deleted, leaving the clearance area.

9.1.9 Land-to-Conductor Transition - Type 1, Type 2, and Type 5 Flex Area The transition between a conductor and a supported or unsupported land should be strain-relieved to prevent broken conductors at this transition interface. An adequate strain-relief can be attained in several ways, depending upon the design density. The primary design requirement is that the transition should not be on or within the coverlayer access hole diameter unless the land is filleted or of a teardrop shape, as shown in Figure 9-1. There are three acceptable designs (see Figure 9-1).

9.2 Holes Alternative hole formation in flexible printed wiring may use die punched, laser cut, cluster punched, plasma etched, and chemical etched holes. Designers should consult fabricators for best process for application and hole size.

9.2.1 Unplated Component Holes When a single-sided flex circuit is used for through-hole mounted component leads, the hole size is typically 0.25 mm to 0.5 mm larger than the component lead diameter.

9.2.2 Plated Component Holes When PTHs are used for component leads, the finished hole size is typically 0.25 mm to 0.5 mm larger than the component lead diameter.

9.2.2.1 Etchback For Type 3 or Type 4 flexible printed wiring, etchback, if required, **shall** be specified on the master drawing.

9.3 Coverlayer Access Openings

9.3.1 Unsupported Holes Access openings in coverlayers to expose unsupported (non-PTH) lands should be at least 0.25 mm smaller than the land to assure "pad capture" by the coverlayer. It is recommended to capture the pad on at least two locations (see Figure 9-2).

9.3.1.1 Coverlay Access Spacing (Unsupported) The limitation for the web left between two adjacent access

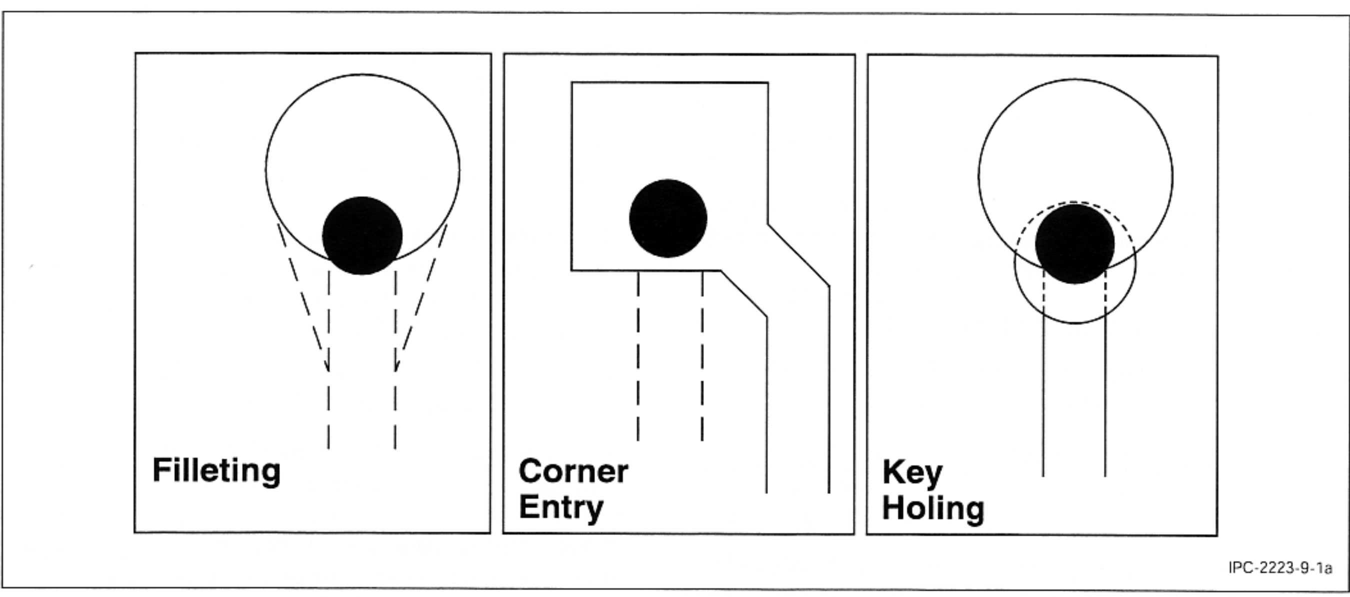


Figure 9-1 Conductor to Land Transitions

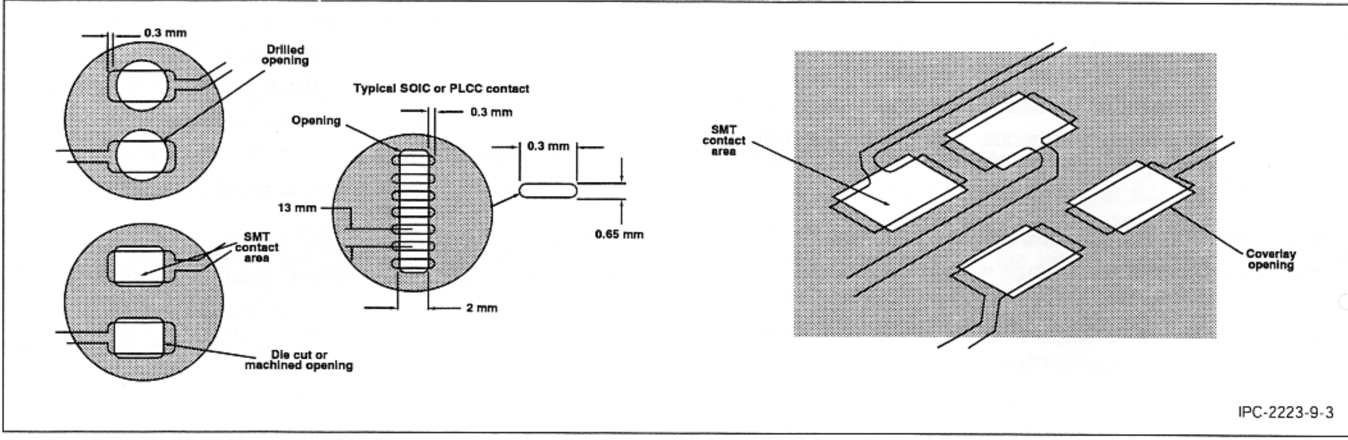


Figure 9-2 Coverlay Access Openings and Exposed Unsupported Lands

openings is 0.25 mm for both die cut and drilled coverlays. Alternative processing is required for spacings <0.25 mm (i.e., windows covering multiple features, laser skiving, photoimagable coatings).

9.3.2 Coverlay Access Openings for Supported Holes Access opening in coverlayers to expose supported PTH lands should be at least 0.25 mm larger than the land to allow for registration tolerances and adhesive squeeze-out (see Figure 9-3 and Figure 9-4).

9.3.2.1 Coverlay Access Spacing (Supported) The limitation for the web left between two adjacent access openings is 0.25 mm for both die cut and drilled coverlays. Alternative processes are required for spacing <0.25 mm (i.e., windows covering multiple features, laser skiving, and photoimageable coatings).

9.3.2.2 Land Access Access to lands may be accomplished by windows (cutouts) or access holes in the coverlay. Windows may be used, and are preferred, if they expose only the lands, because conductor paths exiting from under the coverlay into a window area tend to break at the edge of the coverlay.

The tolerance on the perimeter, windows in the coverlayer, and other features, such as holes >6 mm diameter and slits, **shall** be ≥1 mm. Tighter tolerances down to 0.4 mm may be used when necessary. Access holes and windows **must** be sized such that a 0.13 mm minimum annulus of the land remains exposed through the access under all tolerance conditions. These tolerances include location of the access holes or window and the potential 0.025 mm/mm shrinkage that may occur as a result of etching. Coverlay access holes should be sized to the annular ring requirements of IPC-2221. There are various techniques for access through a coverlayer, as illustrated in Figure 9-2, Figure 9-3, and Figure 9-4.

9.3.2.3 Exposed Lands Where a group of lands will be exposed by a cutout in the coverlayer, dimensions locating

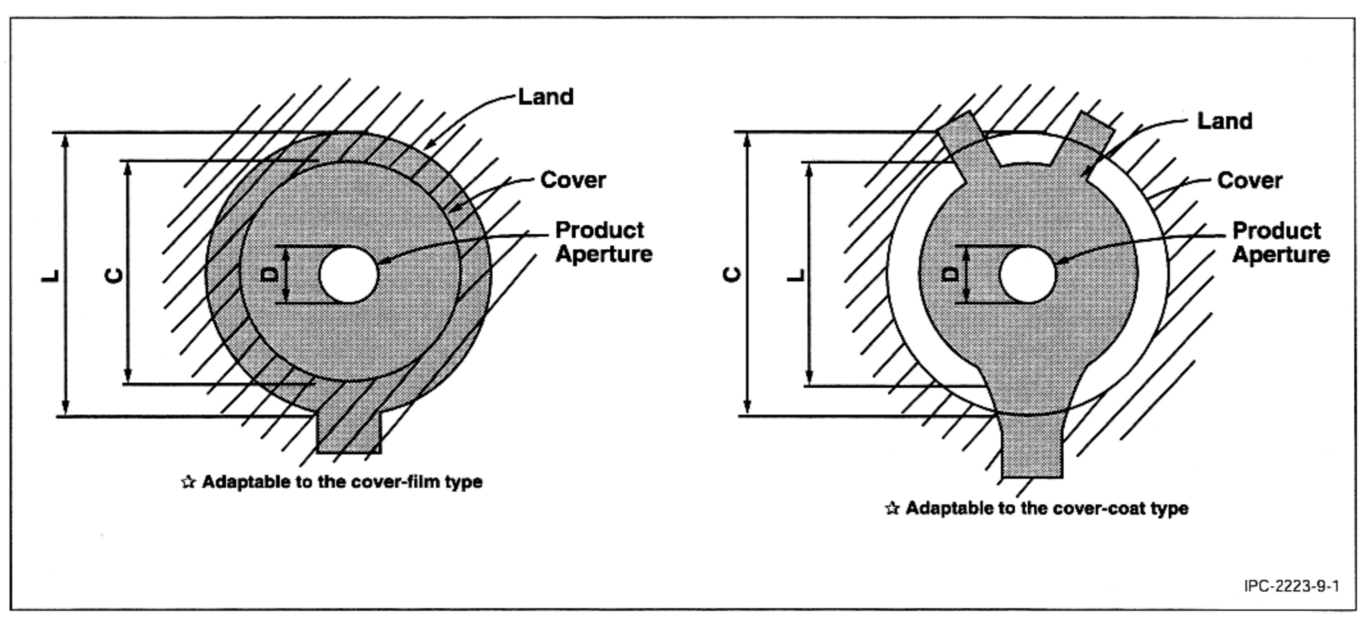


Figure 9-3 Covercoat Opening

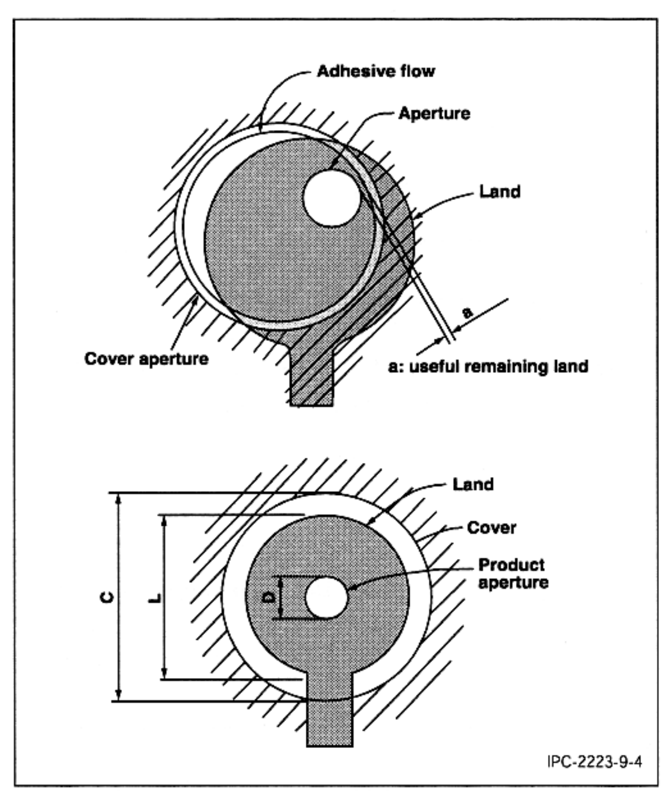


Figure 9-4 Coverlay Access Openings and Exposed Supported Lands

cutout edges with respect to conductors should carry a larger tolerance to allow for bowing of the coverlayer edge.

Large cutouts in coverlayer film extending across a number of conductors may create a stress concentration at the cutout edges, which can cause conductors to break if the circuit is subject to repeated bending, as from vibration in that area.

10 GENERAL CIRCUIT FEATURE REQUIREMENTS

10.1 Conductor Characteristics Conductor characteristics **shall** be in accordance with IPC-2221 and as stated in 10.1.1 and 10.1.2.

Note: It is preferred that all conductor routing in flex areas should have a radius to reduce stress conditions.

10.1.1 Conductor Routing Conductors **shall** not exit from a rigid to flexible section at an angle. Conductors should not pass through a designed bend zone at an angle. Minimum spacing of an angular conductor to a bend or rigid/flex transition should be no less than 5 mm (see Figure 10-1).

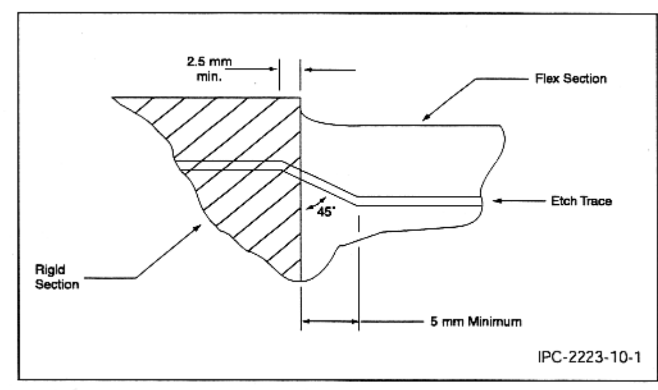


Figure 10-1 Conductor Routing

10.1.2 Edge Spacing Except for edge-board contacts, the minimum distance between conductive surfaces and the edge of the finished board **shall** not be less than the minimum spacing specified in IPC-2221 plus 0.4 mm. Flexible printed wiring that slides into guides **shall** have a minimum

external conductor to guide distance of 1.25 mm or the minimum electrical clearance (see IPC-2221), whichever is greater. Special design applications in areas such as high voltage, surface mount, and radio frequency (RF) technology may require variances to these requirements. Ground and heat sink planes may extend to the edge when required by design. For tighter edge distance, alternative tooling can be used.

10.2 Land Characteristics Land characteristics **shall** be in accordance with IPC-2221.

10.3 Large Conductive Areas Unless otherwise specified, large conductive areas on external or internal layers **shall** be in accordance with 5.2.6 and 5.2.7.

10.3.1 Large External Conductive Areas External conductive areas that extend beyond a 25 mm diameter circle should contain etched areas that will break up the large conductive area but will retain the continuity and functionality of the conductor. Large conductive areas should, if possible, be on the component side.

10.3.2 Large Internal Conductive Areas When a conductive area that extends beyond a 25 mm diameter circle is used on an internal layer, the layer should be placed as near to the center of the circuit as possible. It should also contain etched areas that will break up the large conductive area but retain the continuity and functionality of the conductor. If more than one layer has a large conductive area, the layers should be located in the circuit to provide balanced construction.

11 DOCUMENTATION

Documentation **shall** be in accordance with IPC-2221.

12 QUALITY ASSURANCE

Quality assurance **shall** be in accordance with IPC-2221.



ANSI/IPC-T-50 Terms and Definitions for Interconnecting and Packaging Electronic Circuits Definition Submission/Approval Sheet

The purpose of this form is to keep current with terms routinely used in the industry and their definitions. Individuals or companies are invited to comment. Please complete this form and return to:

IPC
2215 Sanders Road
Northbrook, IL 60062-6135
Fax: 847 509.9798

SUBMITTOR INFORMATION:

Name: ______________________

Company: ______________________

City: ______________________

State/Zip: ______________________

Telephone: ______________________

Date: ______________________

❑ This is a **NEW** term and definition being submitted.
❑ This is an **ADDITION** to an existing term and definition(s).
❑ This is a **CHANGE** to an existing definition.

Term	Definition

If space not adequate, use reverse side or attach additional sheet(s).

Artwork: ❑ Not Applicable ❑ Required ❑ To be supplied
❑ Included: Electronic File Name: ______________________

Document(s) to which this term applies: ______________________

Committees affected by this term: ______________________

Office Use

IPC Office

Date Received: ______________________
Comments Collated: ______________________
Returned for Action: ______________________
Revision Inclusion: ______________________

Committee 2-30

Date of Initial Review: ______________________
Comment Resolution: ______________________
Committee Action: ❑ Accepted ❑ Rejected ❑ Accept Modify

IEC Classification

Classification Code • Serial Number

Terms and Definition Committee Final Approval Authorization:

Committee 2-30 has approved the above term for release in the next revision.

Name: ______________________ Committee: IPC 2-30 Date: ______________________



ASSOCIATION CONNECTING
ELECTRONICS INDUSTRIES®

2215 Sanders Road, Northbrook, IL 60062-6135
Tel. 847.509.9700 Fax 847.509.9798
www.ipc.org

ISBN #1-580981-83-6